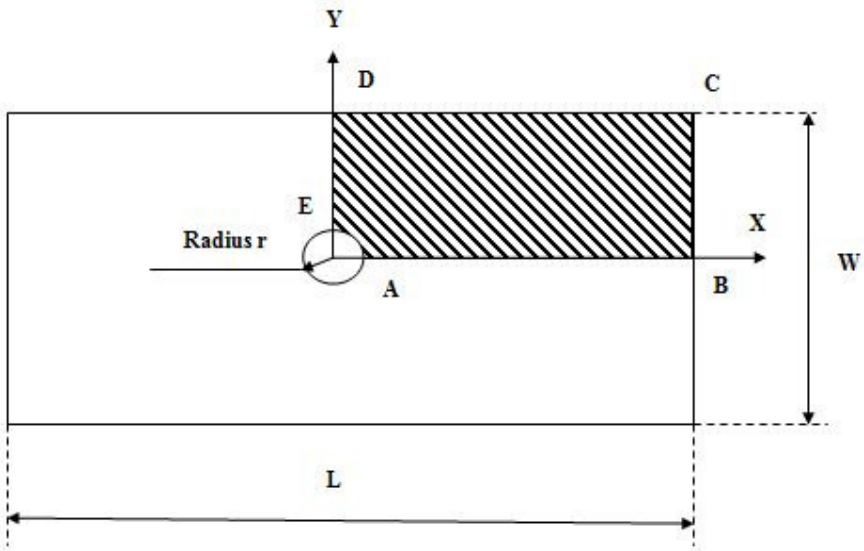
Static analysis of hole in a plate using plain stress and glue element

1. Static analysis of hole in a plate using Plane Stress and Glue Element



Geometric Properties

Thickness = 1*mm*
Length, L= 200*mm*
Width, W = 80*m*
Hole radius, r = 10*mm*

Boundary condition

Along AB, zero Y displacement
Along DE, zero X displacement

Material properties

E=200x10^3*MPa*
v=0.3

Loading condition

Uniform edge load of 1*N/mm* in X direction

PROCEDURE

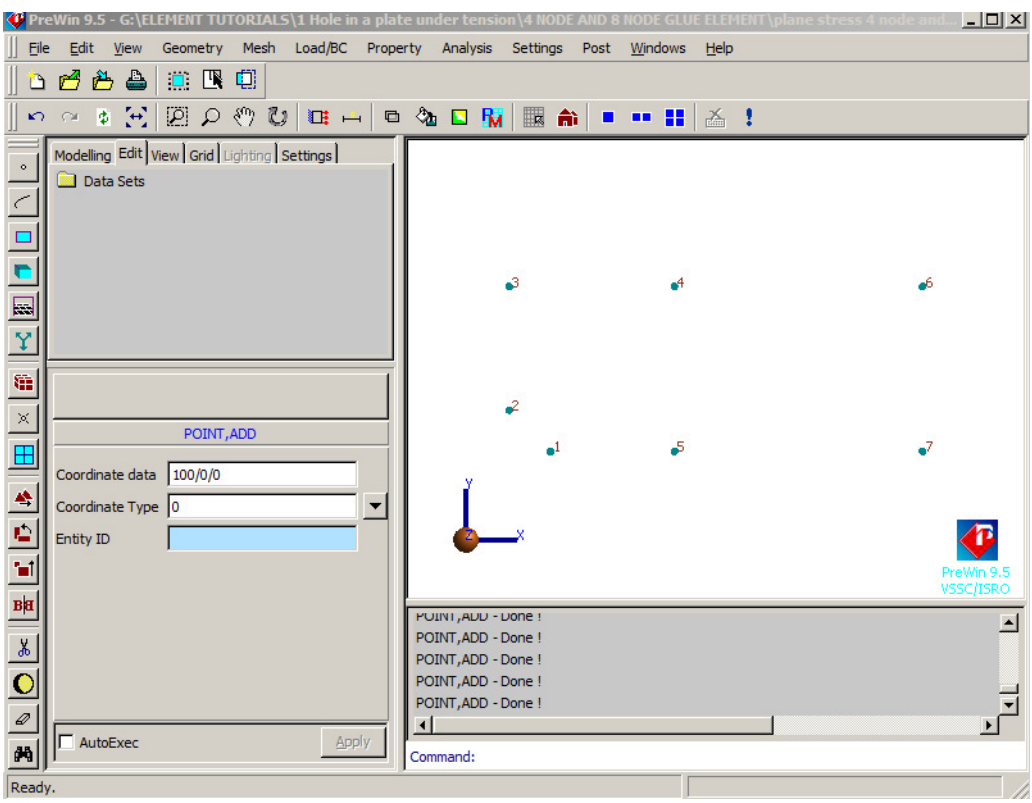
STEP

1. Create points

Create points using the following coordinates

	X	Y	Z
P1	10	0	0
P2	0	10	0
P3	0	40	0
P4	40	40	0
P5	40	0	0
P6	100	40	0
P7	100	0	0

At the end of the above operations, your screen should look like this.



2. Join the points using curves

Command : CURVE, LINE

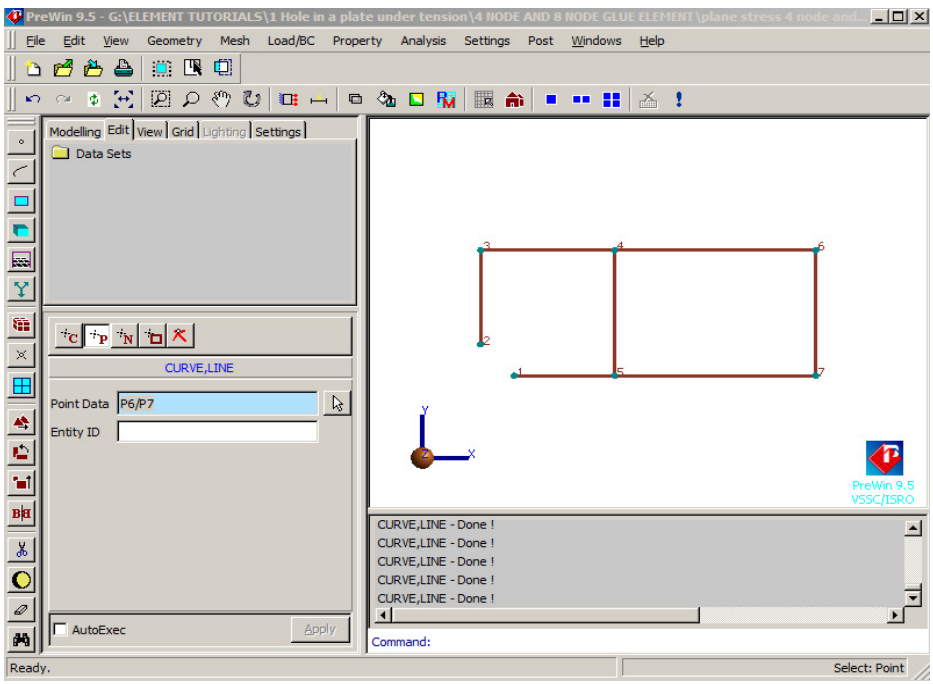
Menu : Geometry → Curve→Create→Line

- Connect all points except points P1 & P2

Parameters

Point Data	P6/P7

At the end of the above operations, your screen should look like this.



3. Create a 2point Arc

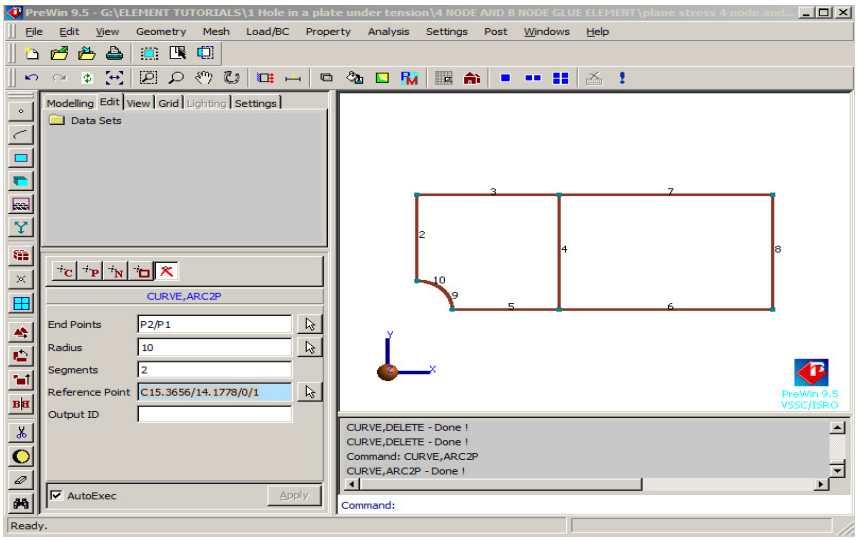
Command : CURVE, ARC2P

Menu : Geometry → Curve→Create→ARC2P

Parameters :

Endpoints	P2/P1
Radius	10
Segments	2
Reference point	A point on the convex side of the arc

At the end of the above operations, your screen should look like this



4. Create surfaces

Command : SURFACE, 2C

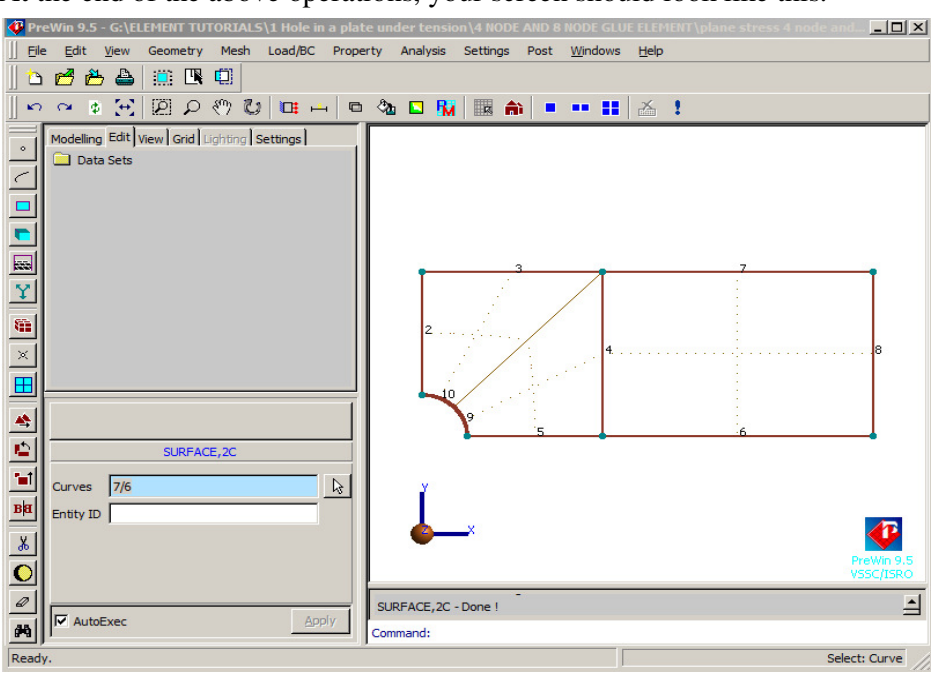
Menu : Geometry → Surface→Create→Using curves→2C

- Select the curve points shown in the above figure, connecting points 10-3, 9-4, 7-6. Points may change according to the selection of curves.

Parameters:

Curves	10/3/9/4/7/6

At the end of the above operations, your screen should look like this.



5. Meshing the surface using Quadratic quadilateral elements

Command : FEM, QUAD

Menu : Mesh → FE Mesh → Generate Quad

Parameters

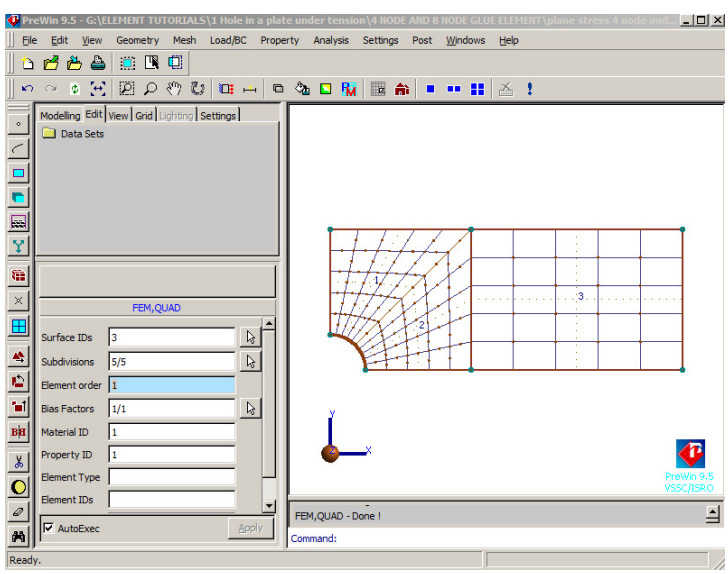
Surface IDS	1/2
Subdivisons	5/5
Element order	2
Bias Factors	1/1
Material ID	1
Property ID	1

6. Meshing the surface using Linear quadilateral elements

Parameters:

Surface IDS	3
Subdivisons	5/5
Element order	1
Bias Factors	1/1
Material ID	1
Property ID	1

At the end of the above operations, your screen should look like this.



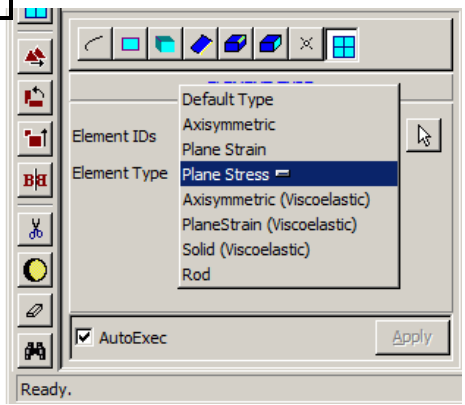
7. Erase surfaces

Command : SURFACE, ERASE

Menu : Geometry → Surface → Miscellaneous → Erase

Parameters :

Surface IDS	1T3



8. Specify Element Type

Command : ELEMENT, TYPE

Menu: Mesh → Element→Modify→Type

Parameters :

Element Type	Plane stress

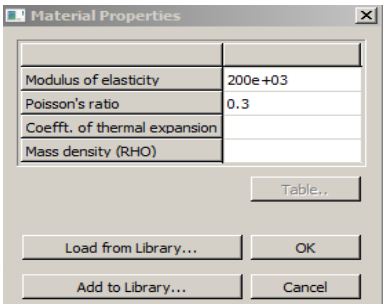
9. Specify material properties

Command : MATERIAL, ADD

Menu: Property → Material → Isotropic → Add

Parameters :

Element IDs	1T75
Material Data	200000/.3/0



*Leave the remaining parameters blank

10.Specify material thickness

Command : THICKNESS, ADD

Menu : Property → Physical → Thickness → Add

Parameters :

Element IDs	1T75
Thickness	1
Set ID	1

11. Add glue element at interfaces of two surfaces\

Command : ELEMENT, GLUE

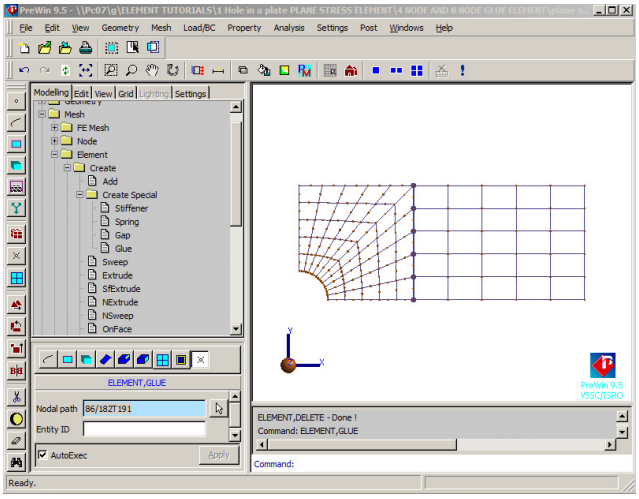
Menu : Mesh → Element →Create →Create Special→Glue

Parameters :

Nodal path	86/182T191

*Leave the remaining parameters blank

At the end of the above operations, your screen should look like this.



12. Specify symmetric boundary conditions

Command : BC, ADD

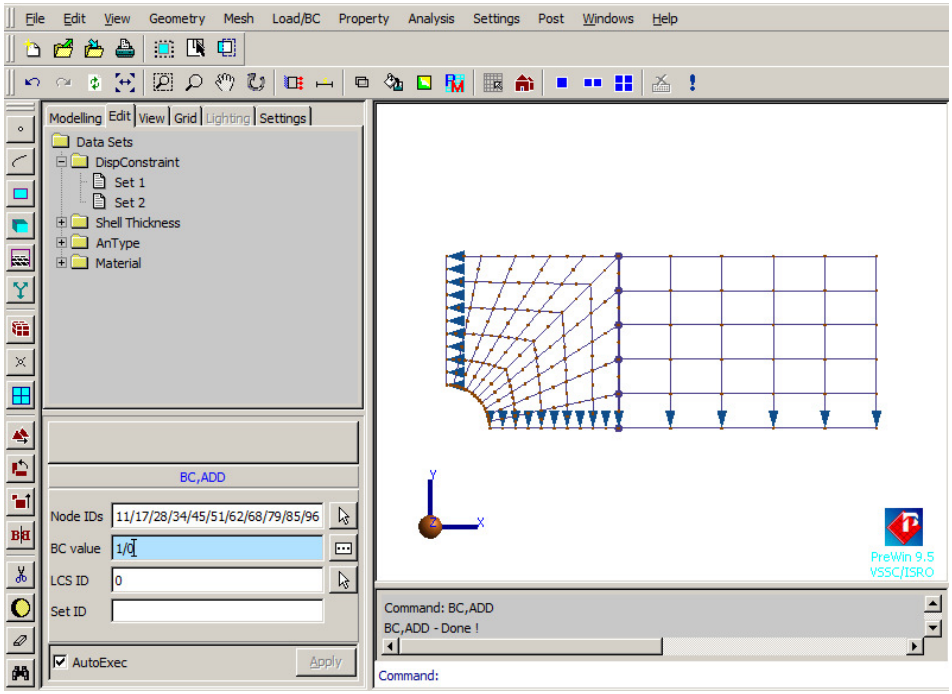
Menu : Load/BC→ Displacement BC →Add

Parameters :

Node IDs	97/108/114/125/131/142/148/159/165/176/182/223T227
BC value	2/0
LCS ID	0

Node IDs	11/17/28/34/45/51/62/68/79/85/96
BC value	1/0
LCS ID	0

At the end of the above operations, your screen should look like this.



13. Specify edge load

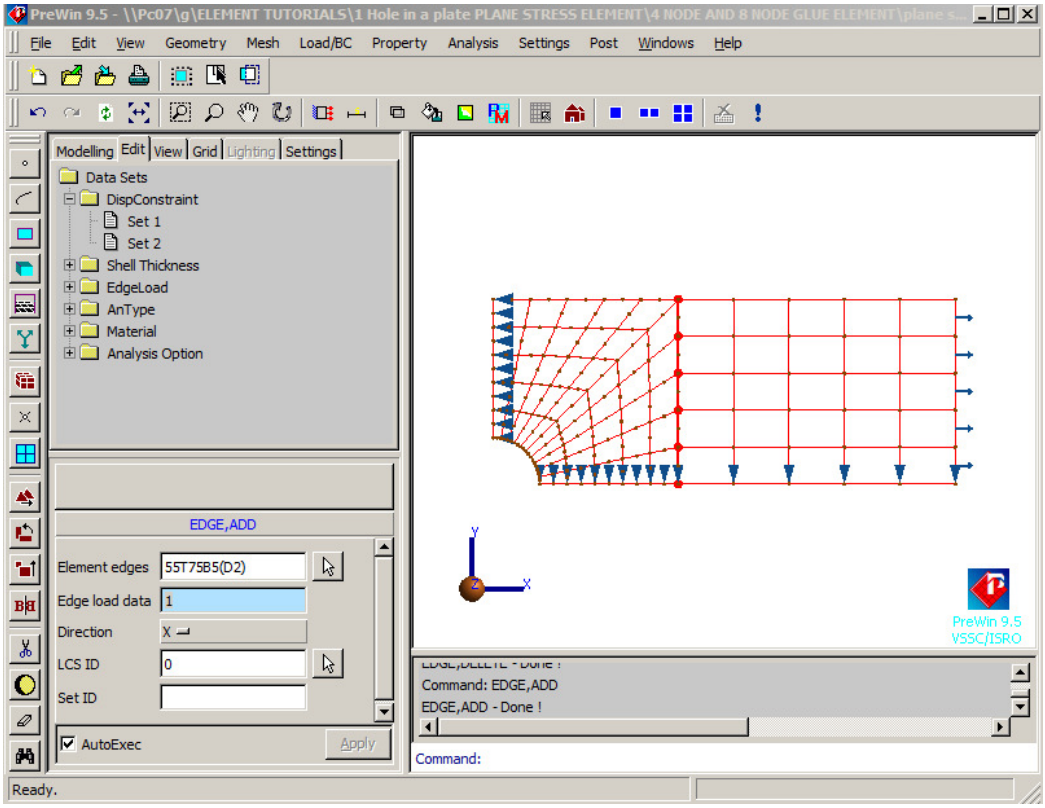
Command : EDGE, ADD

Menu : Load/BC→ Edge load →Add

Parameters :

Element edges	55T75B5(D2)
Edge load data	1
Direction	X
LCS ID	0

At the end of the above operations, your screen should look like this.



14. Set the analysis type

Command : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters

Analysis Type	Static

15. Set the analysis options

Command : ANOPTION, ANOPTION

Menu : Analysis → Analysis Options

Parameters :

Linear Solver	Cholesky

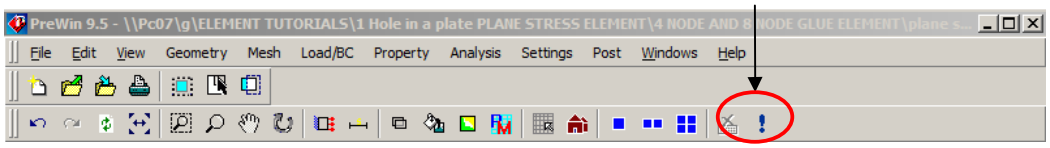
16. Save the project model

Menu : File → Save

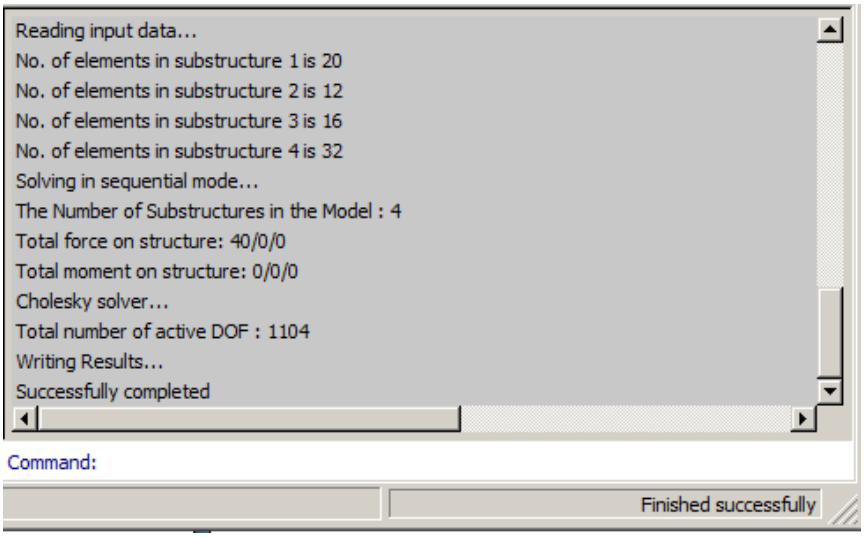
17. Submit the job into FEAST

Menu : Analysis →Run solver

Click here



After the solution is completed the message *"successfully completed"* appears in the *message box*



18. Perform post processing

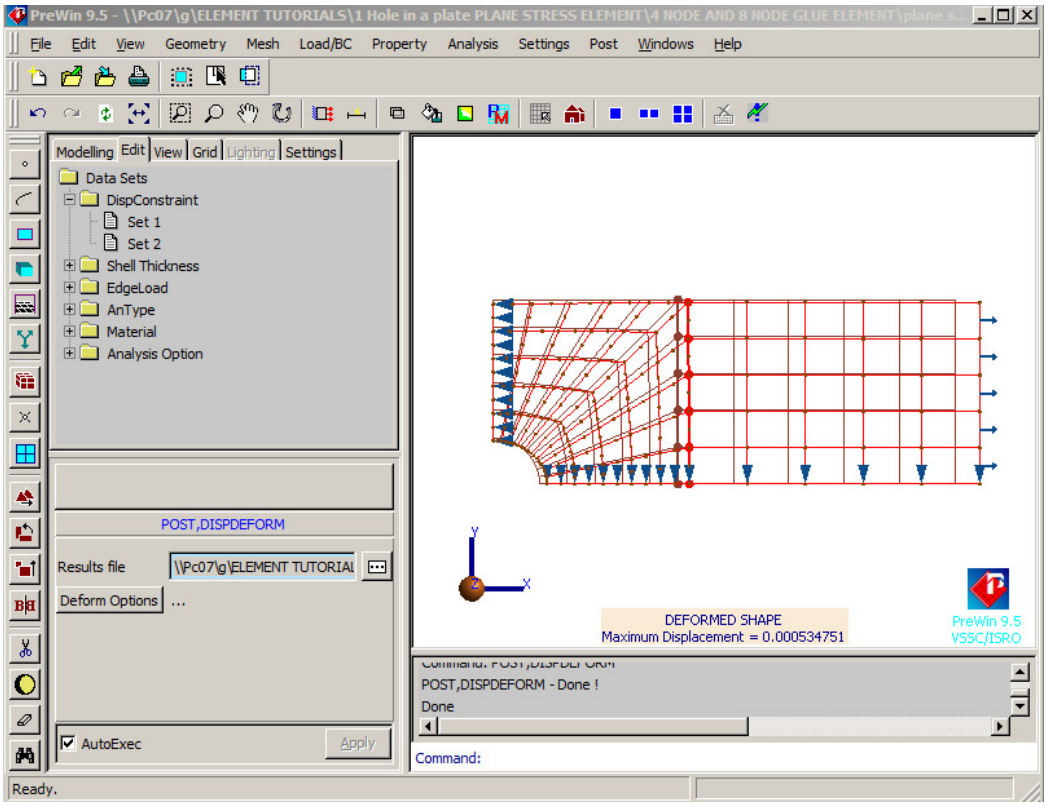
a. Deformation

Command : POST, DISPDEFORM

Menu: Post → Static→Structural deformation → Deformed shape

Results file	Filename. out

At the end of the above operations, your screen should look like this



b. Stress contour

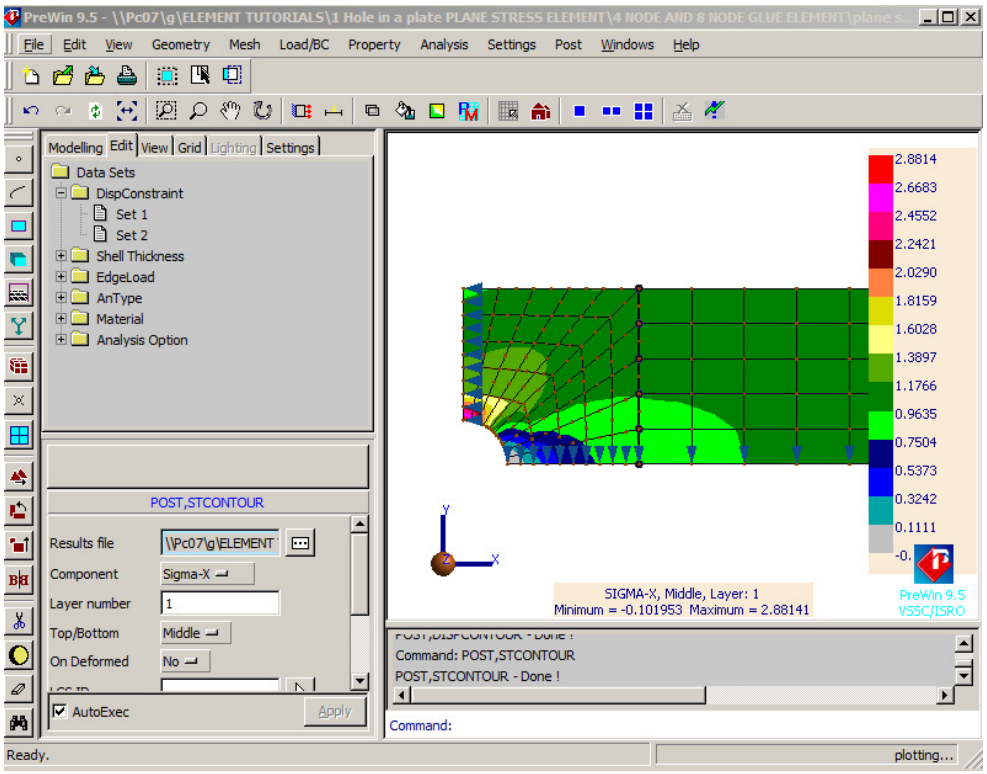
Command : POST, STCONTOUR

Menu : Post →Static →Contour→ Stress

Parameters :

Results file	Filename. out
Component	Resultant
On Deformed	No

At the end of the above operations, your screen should look like this



c. Reaction Force

Command : POST, VRFORCE

Menu : Post → Static→View results →Reaction Force

Parameters :

Results File	Filename. out
Node Ids	97/108/114/125/131/142/148/159/165/176/182/223T227

At the end of the above operations table as shown below.

Reaction Forces

NodeID	FX	FY	FZ	MX	MY	MZ
11	-3.0132	0	0	0	0	0
17	-8.08762	0	0	0	0	0
28	-2.87511	0	0	0	0	0
34	-5.32247	0	0	0	0	0
45	-2.43633	0	0	0	0	0
51	-4.68234	0	0	0	0	0
62	-2.24981	0	0	0	0	0
68	-4.35426	0	0	0	0	0
79	-2.09578	0	0	0	0	0
85	-3.96636	0	0	0	0	0
96	-0.916716	0	0	0	0	0
TOTAL	-40	-1.4e-06	0	0	0	0

Select All | Copy | Delete Column | Delete Row | Close

d. Output can be seen in *.OUT file